Indoor Air 2003; 13: 77–85
www.blackwellpublishing.com/ina
Printed in Denmark.

INDOOR AIR
ISSN 0905-6947

Comparing zonal and CFD model predictions of isothermal indoor airflows to experimental data

Abstract It is inappropriate to use the assumption of instantaneously well-mixed zones to model airflows and pollutant transport in large indoor spaces. We investigate two approaches for describing the details of airflows in large indoor spaces, for accuracy and suitability for integration with multi-zone infiltration models. One approach, called the zonal method, was developed over the last 15 years to provide an improvement over the well-mixed assumption. The second approach is the use of a computational fluid dynamics simulation using a coarse grid model of the large indoor space.

We compare velocity predictions from different formulations of zonal methods and coarse-grid k-ε computational fluid dynamics (CFD) models, to measurements, in a 2D mechanically ventilated isothermal room. Our results suggest that, when airflow details are required, coarse-grid CFD is a better-suited method to predict airflows in large indoor spaces coupled with complex multi-zone buildings, than are the zonal methods. Based on the comparison of pressure predictions from different models, we offer guidance regarding the coupling of a model of detailed airflow in large spaces to algebraic multi-zone infiltration models.

L. Mora[1,2*], A. J. Gadgil[1], E. Wurtz[2]

[1]Lawrence Berkeley National Laboratory, MS 90-3058, Berkeley, CA, USA, [2]L.E.P.T.A.B., University of La Rochelle, av. Michel CREPEAUX, La Rochelle, France

Key words: Indoor air; Computational fluid dynamics; Zonal; Airflow; Simulation.

L. Mora
(Present address)
L.E.P.T.A.B., University of La Rochelle, av. Michel CREPEAU, F-17042 La Rochelle, France
Tel: +33 546 458 785
Fax: +33 546 458 241
e-mail: lmora@univ-lr.fr

Received for review November 2000. Accepted for publication 9 April 2001.

Practical Implications

In several applications it is desirable to model airflows and pollutant flows in complex buildings that contain large indoor spaces such as atriums or large conference halls. For developing such an integrated model, one needs to couple one of the common methods for modeling air and pollutant flows in large complex buildings (e.g., COMIS or CONTAM) with an appropriate model of the large indoor space. This work shows that airflow (and therefore air borne pollutant-flow) predictions in large spaces are substantially more accurate when obtained from a coarse-grid CFD model than from various versions of zonal models. The demand for computer resources remains modest with coarse-grid CFD. This work also discusses the practical problems related to developing such model integration by coupling pressures and airflows between a model of the large indoor space and the building airflow network model.

Introduction

Indoor environmental design requires detailed information about air distribution, such as airflow pattern, velocity, temperature, humidity, and pollutant concentrations. As experimental measurement cannot be a practical design tool, various numerical methods have been developed to simulate these details within the indoor environment. A popular approach of computational simulation is to deploy one of the computational fluid dynamics (CFD) methods. However, solving commonly used turbulence models requires fast computers with large amount of memory. So this approach has mostly been limited to detailed studies of air distribution in single rooms.

Multi-zone infiltration and airflow models such as COMIS (Feustel and Rayner-Hooson, 1990) and CONTAM (Walton, 1997) have been developed to predict airflows in complex buildings. These models are suitable tools to design ventilation systems for complex buildings, as well as to provide necessary inputs for energy analysis tools. They can predict airflows and contaminant transport within the entire building. The applications have usually been based on a strong assumption that the building can be defined as a set of well-mixed volumes or zones of homogeneous composition. While this assumption can be acceptable for small rooms or zones, it becomes unacceptable when modeling large indoor spaces such as atria and auditoria, regarding modeling of phenomena based on local airflows (e.g., drafts, acute or localized pollutant exposures).

The present work is part of a research effort aimed at integrating a detailed model of airflow in large spaces with an algebraic multi-zone infiltration model to describe pollutant transport and coupled airflows within and between complex buildings and large spaces contained within them. Knowledge of airflow is important because advection can contribute significantly to pollutant transport in such buildings.

Over the past 15 years, when zonal models (Bouia and Dalicieux, 1991; Haghighat et al., 2001; Inard, 1988; Li et al., 1998; Rodriguez and Caceres, 1993; Wurtz, 1995) were developed, one of the goal was to obtain an approximate but quicker answer than with CFD models to predict airflow characteristics in large indoor spaces. On the other hand, reducing the number of grid nodes (i.e., using a coarse-grid) in CFD models also permits us to reduce their demand for computational resources to solve airflows in room. Therefore, we compare the ability of both zonal and coarse-grid CFD models to predict airflows in a large indoor space.

In the next section we summarize the requirements imposed by the need to couple a model of large indoor spaces with multi-zone infiltration models. In the third section, we briefly describe zonal methods. In the fourth section, we first present airflow patterns predicted using various zonal models and k-ε CFD models, in a mechanically ventilated isothermal room. Then we present a comparison between velocity predictions from the different formulations of zonal models using the simulation environment SPARK (Buhl et al., 1993), as well as k-ε CFD models, and measurements in the same room geometry published by Nielsen et al. (1978). We also compare the pressure field predictions from the different models. In the last and the fifth section, we summarize our findings and outline directions for future work.

Coupling a large space model into a multi-zone infiltration model

Common usage of multi-zone infiltration models such as COMIS and CONTAM is based on the assumption that state variables except pressures are homogeneous in each building zone (the pressure varies hydrostatically). However this assumption is a very poor approximation for the situation in a large indoor space such as an auditorium or atrium. In order to obtain meaningful predictions of airflow and contaminant dispersion in such spaces contained within complex buildings, it is necessary to integrate a more detailed airflow model of the space into the multi-zone airflow model.

As commonly implemented, multi-zone infiltration models treat each building zone as a single node, and solve the coupled non-linear algebraic system of equations describing airflows in the whole building, relying on the description of flow elements interconnecting the zones. The models treat air as incompressible with variable density. The flow elements connecting the zones, such as cracks or apertures, are described by algebraic relations between the mass airflow rate and the difference of pressure across the element. The pressure variables in such multi-zone infiltration models have the same meaning as in ordinary building science and physics. This meaning (and variable values) must be consistently used in the simplified airflow model of large indoor space, for consistent and successful integrated solution of the coupled problem of airflow in multi-zone building with a large space.

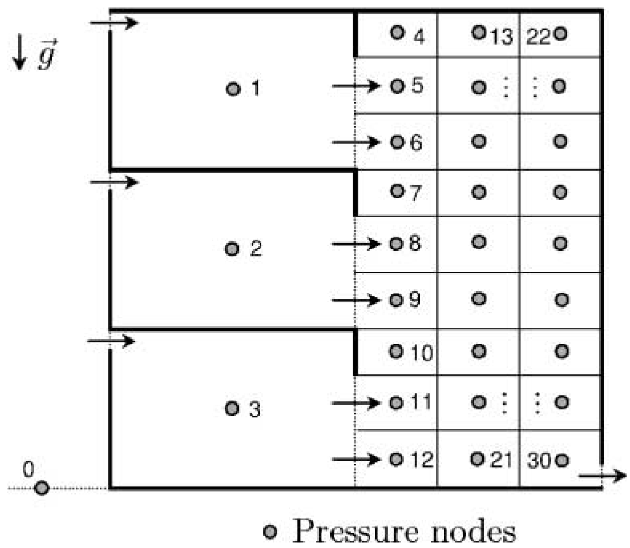

Fig. 1 Section of a multi-zone building

For example, consider a schematic section of an illustrative three-story building composed of three rooms, one on each floor, connected to an atrium by doorways (see Figure 1). In this case a multi-zone infiltration model would compute pressure nodes from 1 to 3, while a model to predict airflows and pressures would be applied to the atrium to calculate pressure nodes 4 through 30. The pressure node 0 is the reference external pressure. The coupling (of both pressures and airflows) between the two models at each doorway location should allow the models to provide a single self-consistent prediction for the entire building.

Zonal models

Common practice

Bouia and Dalicieux (1991) and Wurtz (1995) initiated the development of zonal methods based on solving the pressure field to predict airflow and temperatures in large indoor spaces. In the zonal method, the room is subdivided into a number of control volumes or cells in which temperature and density are assumed to be homogeneous, while pressure varies hydrostatically. Mass and thermal energy balances are applied to each cell, with air treated as an ideal gas. The model of one-way airflow between adjacent cells is based on methods used for large openings in ducts. In these methods, the mass flow rate $\dot{m}_{i,j}$ through an interface of width l, limited by elevations z and $z + \mathrm{d}z$ connecting cell i and cell j is assumed to be governed by a power-law equation as:

$$\dot{m}_{i,j} l \mathrm{d}z = C\rho S(\Delta P_{i,j})^n l \mathrm{d}z \qquad (1)$$

where $\Delta P_{i,j} = (P_i - \rho_i g z_i) - (P_j - \rho_j g z_j)$. It appears that a value for C of 0.83 m/sPa^{-n} for the whole grid except for the apertures and $n = 0.5$ is the common practice (Wurtz et al., 1999). Also, the thermal energy flow is determined using a convection–diffusion relationship across the surface between two cells. This class of models will be called power-law models (PL).

Recently, Voeltzel (1999) applied this approach to predict airflow patterns and temperature field in atria. For this purpose, she incorporated accurate solutions of radiative exchanges between indoor surfaces and solar gains into a zonal model. For airflow modeling, she used a standard set of power-law flow equations such as Equation 1. She obtained good agreement between time-dependent predictions and measurements of temperature. For experiments, she used a 5.1 m-high highly glazed room (ENTPE – SunCell) to validate her zonal model. Temperatures were measured every minute along the vertical centerline of the room at four different heights for 56 h. Time-dependent temperature predictions demonstrated satisfactory agreement with measurement at these four locations. A zonal model also gave more accurate temperature predictions than a one-node model.

In a concurrent and separate research effort, Wurtz et al. (1999) pointed out that increasing the number of cells (i.e., the spatial resolution) in a zonal model does not improve the velocity predictions of the model. To improve the velocity predictions, one must add new laws into the model. Classical models such as Equation 1 cannot adequately represent high velocity regions (e.g. air jets or thermal plumes), because of the inadequate representation of momentum conservation (by approximating it with a relation between mass flow rate and difference of pressure developed for flow across apertures).

Inard (1988) developed an innovative approach to address the inability of the standard zonal method to adequately represent jets and plumes. In order to study the coupling between the thermal plume from a radiator and the airflow in the rest of the room, he patched on to the room model a region for the plume, in which airflow and temperature were known functional relations from textbook idealizations of wall thermal plumes. He, his colleagues, and others extended his method to incorporate free jets, wall jets, and boundary layers in the airflow within the room. Of course, the modeler is presumed to know which specific driven-flow idealization to incorporate into the model in each spatial region. This class of zonal models will be identified in this paper as power-law models with specific driven flows (SDF), or PL-SDF models. In the PL-SDF class of models, Bouia developed an integrated tool called SAMIRA (Bouia, 1993). Within a few years, Wurtz et al. (1999), Musy (1999) and Musy et al. (2001) developed a library of models within the object-oriented simulation environment SPARK. Wurtz's description allows bi-directional flows across common surfaces shared by cells, while Musy developed an automatic generator of zonal models for complex multi-zone buildings, and integrated new libraries into the zonal model for modeling pollutant transport in the room air, radiative heat transfer between inner surfaces, as well as integrating a finite difference model of conduction heat transfer model through the building envelope.

Inard et al. (1996) presented results (obtained with SAMIRA) demonstrating good agreement between experimental data and predictions of temperatures fields under natural and mixed convection using PL-SDF models. The steady-state natural convection experiment is a 3.1 × 3.1 × 2.5 m cell (CETHIL-MINI-BAT test cell), where five wall surfaces are maintained at constant temperature and the sixth surface is in contact with a climatic chamber, allowing control of its surface temperature from −10 to +40°C. Temperature measurements were collected in 200 locations, with 50 sensors in the central vertical plane. Isotherms predicted by zonal models present a good agreement with isotherms constructed from interpolating measured data in this central plane of the cell. Three steady-state mixed convection cases were investigated (one each based on an electric heater, a hot water radiator, and a hot water floor heater) in a ventilated room. Temperature predictions were compared with measurements at seven heights along a vertical line in the central plane of the room. This study presents good agreement with experimental data, and highlights the necessity of using an idealized flow model to describe the thermal plumes generated by radiators and heaters. Musy (1999) also demonstrated the ability of this class of models to predict temperature fields for various heating or cooling systems.

Finally, Lepers (2000) presents good agreement between temperature predictions and measurements in a non-isothermal ventilated room using SAMIRA. The experimental facility is a full-scale room (7.31 × 2.48 × 2.44 m-high) designed by Zhang et al. (1992), in which temperature and horizontal velocity component were measured with a thermocouple and a hot wire probe, respectively, at 205 locations in the central vertical section. Although velocity predictions are about two or three times lower than experimental data in the major part of the simulated room, the airflow pattern is qualitatively well-represented.

Note that in the zonal methods of PL class, what is termed as pressure at each cell is a variable internal to the model with no realistic values, certainly with no relationship to the physically measured pressures within the space. In the next section, we show that PL-class zonal model predict pressures that are greatly in error, and also grid-dependent. This prevents matching the pressures in a COMIS-type infiltration airflow model of a complex building, with those of a PL-class model of airflows within a large space enclosed within that building and in communication with it.

An alternate formulation of zonal models

Axley (2001) recently proposed a method to overcome a major shortcoming of the PL class of zonal models. When a PL-class zonal model is applied to predict airflow through a room, the total predicted pressure drop across the room depends linearly on the number of cells used. This shortcoming of the zonal approach has been long known to the practitioners, but no remedy had been proposed for this till now, essentially because the use of zonal models was restricted to single zone buildings where pressure consistency was not an important issue.

Axley (2001) proposal avoids the grid dependence of pressure in current zonal models. In this approach one assumes that airflow in rooms is determined by the interplay between pressure drops across and, surface drag on, air in each cell. Then the airflow in all cells can be determined by considering the transfer of shear stresses to the nearest wall surfaces. Applying a momentum balance along a differential conduit (see Figure 2) of height ds and length Δr between the pressure node P_i and the pressure node P_j, of two adjacent cells leads to:

$$\Delta P_{i,j} w \mathrm{d}s = -\frac{\mathrm{d}\bar{\tau}_{sr}}{\mathrm{d}s} w \Delta r \mathrm{d}s \tag{2}$$

Using the Prandtl's mixing length expression of shear stress for turbulent flow, and given a velocity profile along the dimension perpendicular to the nearest wall, the cell-to-cell difference of pressure expression becomes:

$$\Delta P_{i,j} \approx 2k_s \frac{\kappa^2 a^3 \Delta r}{\rho w^2 \Delta s^3} \dot{m}_{i,j}^2 \tag{3}$$

From now, this model will be called the surface-drag model (SD). Like the PL model, it can be augmented by adding specific driven flow formulations in specific

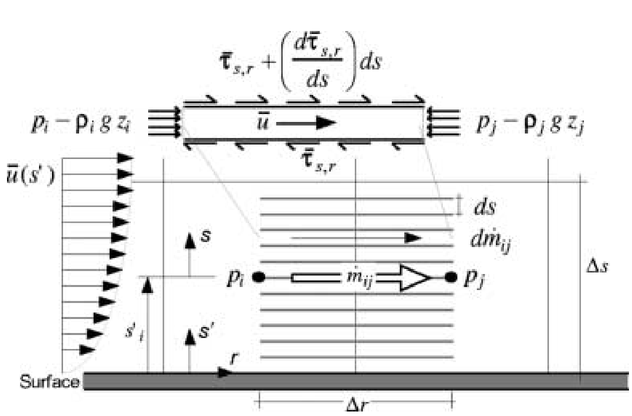

Fig. 2 Surface-drag momentum balance flow model (Fig. 2b in Axley 2001)

regions of space. In this latter case, the obtained SD model will be called SD-SDF, for SD model with specific driven flow.

The next section compares airflow patterns and velocity predictions given by the various formulations of zonal models described above with measurements in a mechanically ventilated isothermal room.

Comparison with Nielsen's experiment

Nielsen (1978) describes velocity measurements in a rectangular parallelepiped scaled model of a room ($H = 89.3$ mm) in which the isothermal airflow is expected to be almost two-dimensional (see Figure 3). The inlet velocity U_{in} is imposed as the Reynolds number Re = 5000 based on inlet slot height ($U_{\mathrm{in}} = 15.02$ m/s). Detailed measurements of velocity profiles are provided along four lines through the central vertical plane located at $y = W/2$: two vertical (at $x = H$ and $x = 2H$), and two horizontal (at $z = 0.972H$ and $z = 0.028H$).

We conducted simulations of airflow in the full-scale geometry ($H = 3$ m) equivalent to Nielsen's experiments, using all four formulations discussed above: PL, PL-SDF, SD and SD-SDF. In the SDF versions, specific equations describe the jet induced by the inlet slot geometry description of Nielsen's experiment. In these conditions, the inlet velocity is imposed as

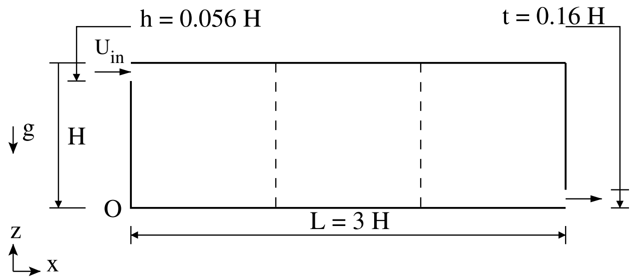

Fig. 3 Description of Nielsen's experiment setup

$U_{in} = 0.447$ m/s. As an alternate simplified method to predict airflows in large spaces we also explored the possibility of applying a coarse-grid conventional k-ε CFD model to this configuration. Zonal model simulations were performed using the object-oriented simulation environment spark and k-ε CFD simulations were performed with the commercial code StarCD.

In this section, we compare predictions of airflow patterns and velocity profiles using the different models discussed above, as well as the ability of each class of models to predict the total pressure drop across the test room (i.e. from the inlet to the outlet). The pressure drop across the room is directly relevant to the model's suitability for integration with a COMIS-type model for multi-zone airflow in complex buildings.

Airflow patterns

Power-law model. For the results presented here, $C = 0.83$ and $n = 0.5$ in Equation 1. The results of airflow predictions with the classical (i.e., PL) zonal model, are presented in Figure 4. We see that the predicted airflows are unidirectional (there is no recirculation), and there is no wall jet predicted. The airflow is spread uniformly across the vertical section of the room.

We then added a specific driven flow model to the classical PL model to describe the wall jet downstream the inlet slot. This jet model is the well-established isothermal wall jet model developed by Rajaratnam (1976). The predictions of this PL-SDF model are shown in Figure 5. The entrainment of room air into the wall jet is not clearly predicted, nor is recirculation of room air induced by the wall jet. The wall jet seems to bounce off the wall opposite the entrance slot and drives a weak recirculation in that region.

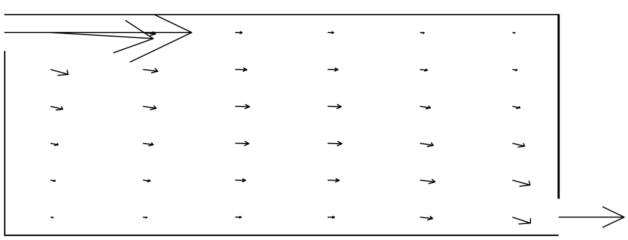

Fig. 4 Airflow pattern predicted by the PL model

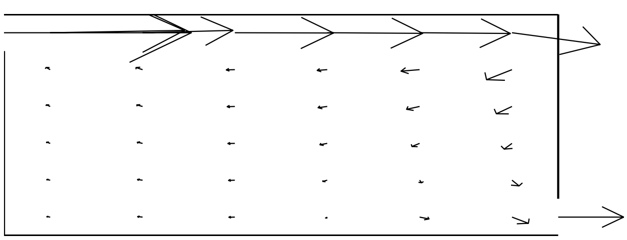

Fig. 5 Airflow pattern predicted by the PL-SDF model

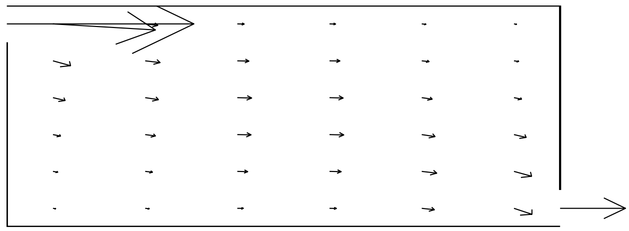

Fig. 6 Airflow pattern predicted by the SD model

Surface-drag model. The airflow pattern predicted with the SD formulation (see Figure 6) is quite similar to the PL model predictions presented in Figure 4. There is no dominant flow in the room, nor any recirculation induced by the interaction of the jet with the enclosure walls. This SD model is identical to that described by Axley (2001), except that Axley used CONTAM (Walton, 1997) to calculate the solution whereas we used the SPARK engine for this purpose. Then we patched the wall jet model developed by Rajaratnam (1976), into this SD model. The predictions from this SD-SDF formulation are shown in Figure 7, and are very similar to Figure 5 for PL-SDF model.

The k-ε CFD model. We performed airflow simulations in the test case geometry using a conventional k-ε CFD model, using different mesh sizes, ranging from 6×6 to 40×40. Our intention was to characterize predictions from coarse-grid CFD, and compare these with experiment and predictions from various zonal methods. Only for the case of the 40×40 grid we refined the mesh near wall surfaces to ensure a boundary layer resolution that satisfies the criterion of applicability of wall functions (in this case $y^+ < 40$). In other, coarser, grids the cell sizes adjacent to the walls were set to 15 cm in the direction perpendicular to the wall.

Chen and Weiran (1998) compared CFD predictions using the standard k-ε as well as his newly developed zero-order turbulence models, with Nielsen's experiment. Our 40×40 grid k-ε CFD results agree very well with those of Chen and Weiran (1998) using standard k-ε turbulence model. Figure 8 shows results for a 10×10 grid, and Figure 9 shows those for the 40×40

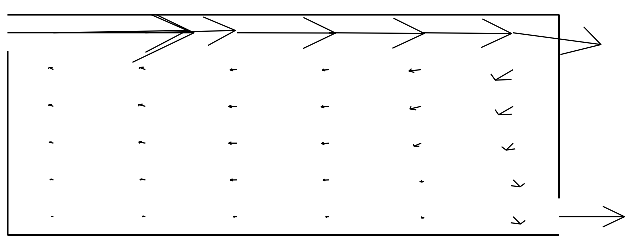

Fig. 7 Airflow pattern predicted by the SD-SDF model

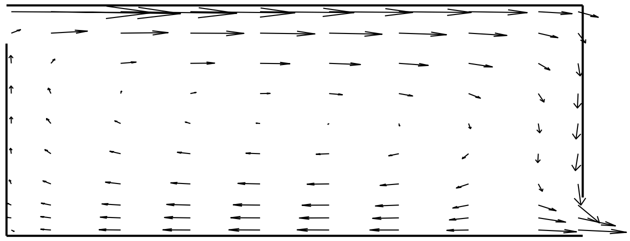

Fig. 8 Airflow pattern for the 10 × 10 grid k-ε CFD model

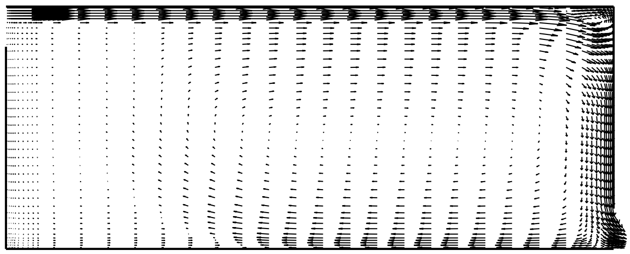

Fig. 9 Airflow pattern for the 40 × 40 grid k-ε CFD model

grid. Both meshes predict a large recirculation loop around the center of the room, and significant entrainment of the room air in the inlet jet. While slight differences among the four zonal formulations do exist, none predict the circulation loop around the geometric center of the room; this is true even for those zonal models for which the specific driven flow model patch predicts the jet itself. The small reverse flow predicted by PL-SDF and SD-SDF models (see Figures 5 and 7) puts the center of recirculation near top of the room, just below the jet. The next section presents details of the velocity predictions from the different models, and compares them with experimental data.

Velocity profiles

A comparison of velocity predictions by different zonal models with experimental data along the vertical line at $x = 2H$ is presented in Figure 10. SDF versions of zonal model do predict some recirculation, but with a peak typically only about 10% of the experimentally observed peak recirculation flow. The air velocities in the wall jet region are well-predicted by specific driven flow (PL-SDF and SD-SDF) models (Figure 10b) but none of the four zonal model formulations is able to reasonably predict either the geometry or the magnitude of the recirculation. Note that the recirculation is seen as negative air velocity values below about $z/H = 0.6$ in Figure 10. In addition, velocity predictions are not significantly improved by using the newer SD formulation in place of the older PL. Velocity predictions with the four zonal model formulations compare equally poorly with experimental results at other sections of the room: the vertical line at $x = H$, and two horizontal lines, one at $z = 0.972H$ (through the air inlet) and the other at $z = 0.028H$ (through the air outlet). These comparisons are not shown for brevity. As earlier pointed out by Wurtz et al. (1999), even substantially increasing the number of cells in the zonal models will not improve the velocity predictions. Thus the predictions obtained with the 6 × 6 cells are representative of those obtained with larger number of cells for the respective formulation of the various zonal models.

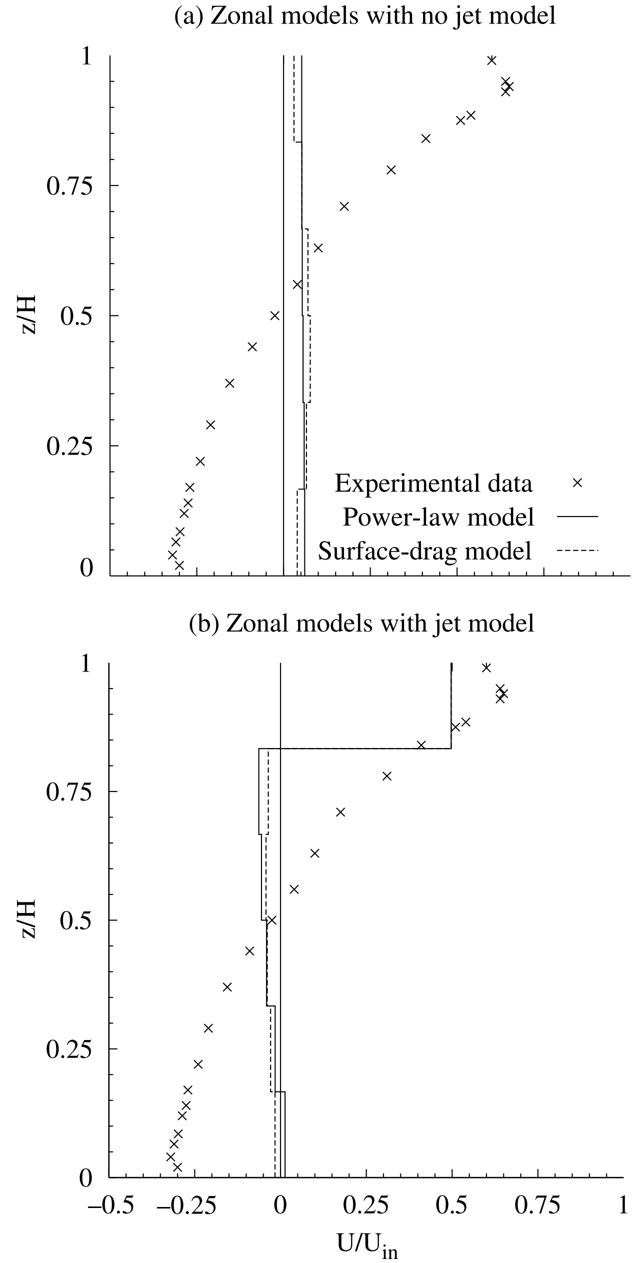

Fig. 10 Comparison of velocity profiles predicted by zonal models with experimental data, in the center section at $x = 2H$

The comparison of velocity predictions with coarse-grid CFD model is shown for all the four sections of the room mentioned above: the vertical lines at $x = H$

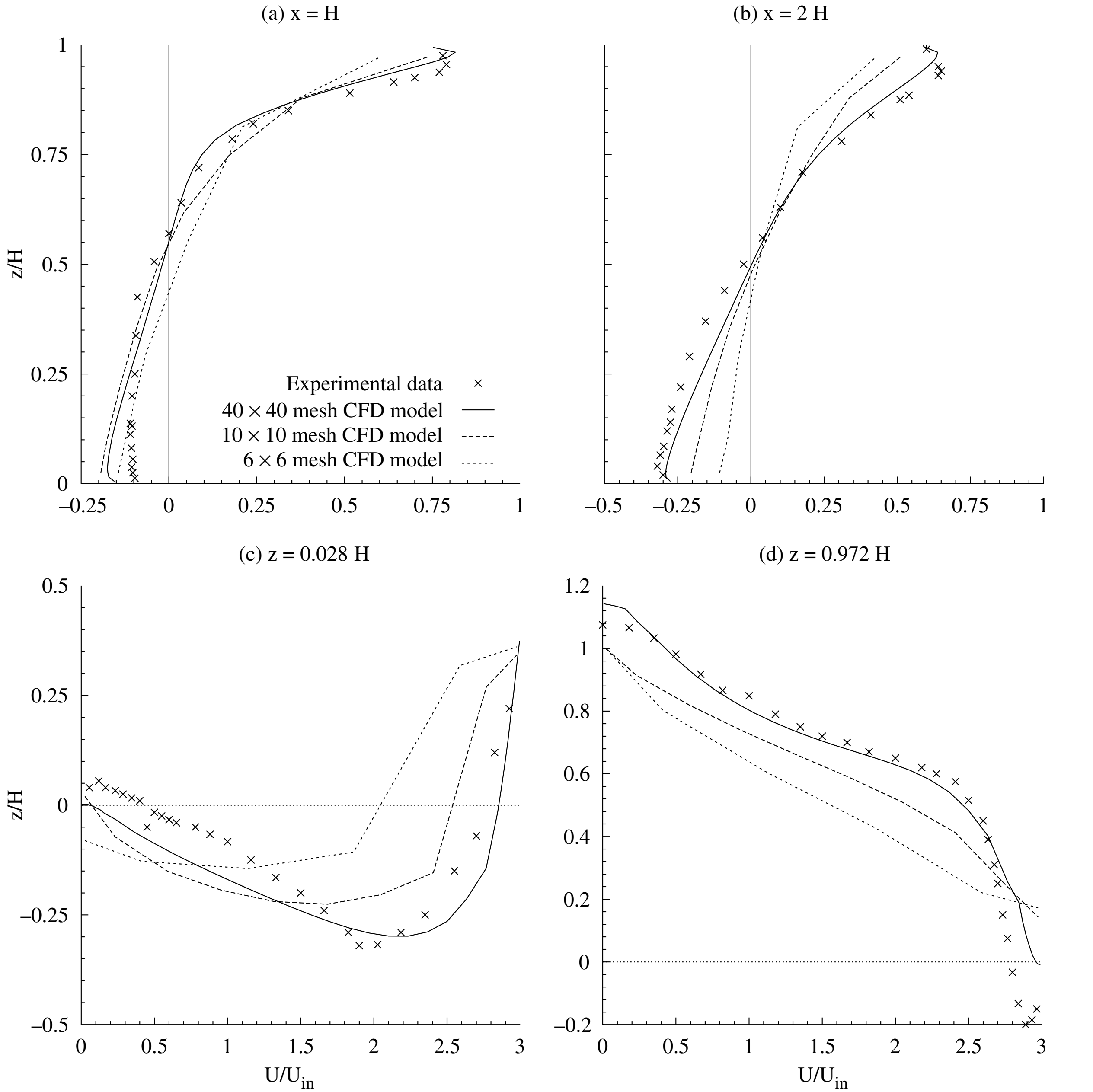

Fig. 11 Comparison of velocity profiles predicted by CFD models with experimental data in four section of the room: at (a) $x = H$, (*b*) at $x = 2H$, (c) at $z = 0.028H$ and (d) at $z = 0.972H$

and $x = 2H$, and the horizontal lines $z = 0.972H$ (through the air inlet) and $z = 0.028H$ (through the air outlet), in Figure 11. In this figure, we compare k-ε CFD model predictions for velocities, based on 6×6 and 10×10 grids, to predictions using 40×40 grid and experimental data. Compared with measurements, we see that all simulations underestimate the recirculation. The results of the 6×6 and 10×10 grids show a jet decay that is slightly too rapid, but on the whole coarse-grid predictions give satisfactory agreement with the experiment. Unlike the case with zonal models, successive increases in the number of cells lead to successive improvements in the predictions of coarse-grid CFD models – until highly resolved grid-independent results are reached.

These results suggest that coarse-grid conventional k-ε CFD model is a good candidate for simplified predictions of the details of airflows, and consequently of contaminant transport, in large spaces connected to complex buildings. Also, this approach offers a satisfactory agreement with the experimental data in the jet region without any expert knowledge to patch an idealized wall jet velocity formula into the computational space at the correct location.

Pressure predictions

Correct prediction of the pressure field is vital for integrating an airflow model of large space into multi-zone airflow models. Although the test case we chose has been widely studied, we were unable to find pressure data in the literature. In one case, where researchers have conducted detailed CFD simulations of airflow in this geometry with Large Eddy Simulation, we found that the pressure field files had been discarded because there were thought to be of little interest. Experimentally, it may well be impossible to measure pressure drops across the room in this geometry at this flow rate because the pressure drop is smaller than the detection limit of available research instrumentation.

Zonal (PL and SD formulations) and the k-ε CFD models were applied to different grids to predict the total pressure drop between the inlet region and the outlet region of the test room. The results are summarized in Figure 12. As Axley (2001) pointed out, the power-law (PL) zonal model predicts a total pressure drop across the test room that is linearly dependent on the number of cells used for dividing the room space. The SD formulation, as expected, shows grid independence. It predicts a total pressure drop about six times smaller than the k-ε CFD model result for a 40×40 grid. This large difference is not entirely unexpected; the SD formulation does not account for molecular and turbulent viscous dissipation of momentum in the core of the room. Note that the coarse-grid CFD results are also sensitive to the number of cells used, although the results appear to flatten asymptotically as the number of cells increases.

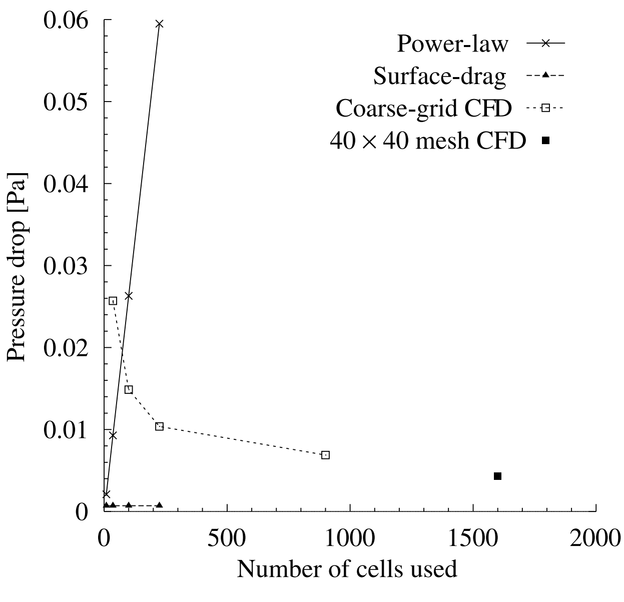

Fig. 12 Total pressure drop across the test room

However, as Musy (1999, Chap. 5) explains, the pressure variable used in zonal models is really an internal variable used to balance the flow equations, not to be confused with the manometric pressure, used as a state variable in fluid mechanics. Thus the quality of pressure predictions by the various zonal models should not be the reason for their acceptance or rejection. If zonal models have to be integrated with a COMIS type model, their pressure predictions should be ignored, and only the airflows should be matched at the common interfaces shared by the two models.

Note that in terms of experimental research instrumentation, the lower detection limit for pressure differences is about 0.1 Pa. The vertical axis in Figure 12, on the other hand ends at 0.06 Pa, much below this detection of limit. In a real building, interzone pressure differences of the order of 10 Pa are common.

Conclusion

Conventional zonal models can estimate airflows, heat and contaminant transport rapidly and with low requirements regarding input data. This was especially appropriate when computers were slow and expensive. However, for accurately modeling pollutant transport in complex buildings, airflows, which contribute significantly to the overall pollutant transport, need to be adequately predicted.

Various formulations of zonal models did not provide satisfactory predictions of airflows under isothermal conditions. Other researchers, (e.g. Lepers, 2000; Wurtz et al., 1999), indicate that such models can predict temperature field and low-resolution details of airflows in non-isothermal conditions. On the other hand, velocity predictions from coarse-grid CFD models are in better agreement with measurements. We note that for these 2D CFD simulations using 10×10 coarse grid, the CPU time required was 3.23 s on a SGI-IRIX workstation (for the 40×40 grid this increased 13 times, to 42 s). The CPU time demand by the coarse-grid CFD calculation does not represent a large computational burden, and this could be reduced further by using the extremely fast solvers developed at Lawrence Berkeley National Laboratory, CA, USA, such as VarDen (Almgren et al., 1998.

Our results suggest that coarse-grid k-ε CFD can be a satisfactory alternative to zonal methods where more accurate details are required, for predicting airflows and contaminant transport in large indoor spaces connected to a complex multi-zone building. In a separate research effort we are addressing acceptable grid-coarseness for satisfactory approximate results and also extending this method to mixed convection configurations.

Acknowledgements

This research was supported by the Office of Nonproliferation and National Security, Chemical and Biological Non-proliferation Program of the US Department of Energy under Contract No. DE-AC03-76SF00098. We would like to thank particularly Elizabeth Finlayson for her generous advice about using the software StarCD, and Dimitri Curtil for his technical help with using SPARK. We would also like to thank Michael Sohn and David Lorenzetti for improving the clarity of this document with their comments.

References

Almgren, A.S., Bell, J.B., Colella, P., Howell, L.H. and Welcome, M.L. (1998) A conservative adaptive projection method for the variable density incompressible Navier-Stokes equations, *J. Comput. Phys.*, **142**, 1–46.

Axley, J.W. (2001) Surface-drag flow relations for zonal modeling, *Building Environ.*, **36**, 843–850.

Bouia, H. (1993) *Modélisation simplifiée d'écoulements de convection mixte interne: Application aux échanges thermo-aérauliques dans les locaux*, PhD Thesis, Poitiers, France, University of Poitiers.

Bouia, H. and Dalicieux, P. (1991) Simplified modeling of air movements inside dwelling room. In: *Proceedings of Building Simulation '91 Conference*, Nice, France, IBPSA (The International Building Performance Simulation Association), 106–110.

Buhl, W.F., Erdem, A.E. and Winkelmann, F.C. (1993) Recent improvements in SPARK: strong component decomposition, multivalued objects, and graphical interface. In: *Proceedings of the Third International IBPSA Conference, Building Simulation '93*, Nice, France, IBPSA, 283–289.

Chen, Q. and Weiran, X. (1998) A zero-equation turbulence model for indoor airflow simulation, *Energy and Buildings*, **28**, 137–144.

Feustel, H.E. and Rayner-Hooson, A. (1990) *COMIS Fundamentals*, Berkeley, CA, Lawrence Berkeley National Laboratory, LBNL–28560.

Haghighat, R., Lin, Y. and Megri, A.C. (2001) Development and validation of a zonal model POMA, *Building Environ.*, **36**, 1039–1047.

Inard, C. (1988) *Contribution à l'étude du couplage thermique entre un émetteur de chauffage et un local. Etudes expérimentales en chambres climatiques*, PhD Thesis, Lyon, France, National Institute of Applied Sciences (INSA).

Inard, C., Bouia, H. and Dalacieux, P. (1996) Prediction of temperature distribution in buildings with a zonal model, *Energy and Building*, **24**, 125–132.

Lepers, S. (2000) *Modélisation des écoulements de l'air dans les bâtiments à l'aide de code CFD. Contribution à l'élaboration d'un protocole de validation*, PhD Thesis, Lyon, France, National Institute of Applied Sciences (INSA).

Li, Y., Delsante, A., Symons, J.G. and Chen, L. (1998) Comparison of zonal and CFD modeling of natural ventilation in thermally stratified building. In: *Proceedings of Air Distribution in Rooms Conference*, Stockholm, (ROOMVENT'98), Vol. 2, 415–422.

Musy, M. (1999) *Automatic Generation of Zonal models to Perform Airflow and Thermal Simulation in Buildings (in French)*, PhD Thesis, La Rochelle, France, LEPTAB – University of La Rochelle.

Musy, M., Wurtz, E., Winkelmann, F.W. and Allard, F. (2001) Generation of a zonal model to simulate natural convection in a room with a radiative/convective heater, *Building Environ.*, **36**, 589–596.

Rajaratnam, N. (1976) *Turbulent Jets*, Amsterdam, Elsevier Scientific Publishing Co.

Rodriguez, E. and Caceres, I. (1993) *Draft Proposal for a Stratification Predictive Model*. Technical report, Commission of European Communities, 1993.

Nielsen, P.V., Restivo, A. and Whitelaw, J.H. (1978) The velocity characteristics of ventilated rooms, *J. Fluids Eng.*, **100**, 291–298.

Voeltzel A. (1999) *Dynamic Thermal and Airflow Modeling of Large highly Glazed Spaces (in French)*, PhD Thesis, Lyon, France, LASH – ENTPE.

Walton, G.N. (1997) *Contam96 User Manual*, Gaithersburg, Building and Fire Research Laboratory, National Institute of Standards and Technology.

Wurtz, E. (1995) *Three-Dimensional Modeling of Thermal and Airflow Transfers in Building Using an Object-Oriented Simulation Environment (in French)*, PhD Thesis, Paris, Ecole Nationale des Ponts et Chaussees.

Wurtz, E., Nataf, J.-M. and Winkelmann, F.W. (1999) Two- and three-dimensional natural and mixed convection simulation using modular zonal models in buildings, *Int. J. Heat Mass Transfer*, **42**, 923–940.

Zhang, J.S., Christianson, L.L., Wu, G.J. and Riskowski, G.L. (1992) Detailed measurements of room air distribution for evaluating numerical simulation models, *ASHRAE Trans.*, **98**, 58–65.